Academic Journal of Science and Technology
ISSN: 2771-3032 | Vol. 20, No. 1, 2026

Analysis of Transient Flow Field in a Medical Composite Plunger Pump Based on CFD

Wenyu Hang, Xingde Liu, Jiarui Liang, Junhan Wang

School of Mechanical and Electrical Engineering, Jilin University of Chemical Technology, Jilin, Jilin, 132021, China

Abstract: This study introduces a medical composite plunger pump and elaborates on its structure and working principle in detail. To investigate the dynamic characteristics of the fluid during the suction and exhaust phases of the medical composite plunger pump, a gap model was constructed based on dynamic mesh technology, enabling coupled simulation of the flow field within the pump and the motion of the check valve spool. Through numerical simulation, the overall flow field velocity, pressure distribution, and pulsation characteristics of the plunger pump were analyzed under a motor speed condition of 140 r/min. The resulting velocity and pressure contour plots of the fluid domain over time facilitated a fluid-structure interaction analysis of the flow field variations within one cycle of the medical composite plunger pump. This approach provides a viable method for further exploring the flow field characteristics of the medical composite plunger pump and offers data support for subsequent optimization of its operating parameters and enhancement of its performance.

Keywords: Compound Pump; Plunger Pump; Check Valve; Dynamic Mesh; Numerical Simulation.

1. Introduction

As a power device integrating positive pressure output and negative pressure suction, the medical composite plunger pump has broad application prospects in medical equipment, precision machinery and other fields due to its compact structure, small size, stable power output and other advantages. The reciprocating motion of the piston is driven by a crank mechanism to realize the integrated integration of positive and negative pressure circuits, solving the problems of complex structure and poor reliability caused by the dual-circuit design required by traditional positive and negative pressure drive systems. However, experiments conducted by our supervisor's team have only verified the basic performance of the air pressure output of this pump under the 24V working condition. The influence mechanism of internal flow field evolution has not been clearly revealed, which limits the precise adaptation of the pump body to scenarios with different power requirements.

Computational fluid dynamics (CFD) technology can intuitively reveal the pressure, velocity distribution and energy loss law of the flow field inside the pump, and serves as a core method for performance analysis and optimization of pump equipment. In this paper, a medical composite plunger pump is taken as the research object. Based on the actual device, a three-dimensional fluid simulation model is established, and targeted simulation conditions are designed. Through comparative verification of simulations, the flow field characteristics inside the pump and their influence on macroscopic fluid performance are studied, providing a theoretical basis for structural optimization, working condition expansion and performance improvement of the pump body, and making up for the lack of in-depth multi-condition flow field analysis of such pumps in existing studies.[1]-[3]

2. Research Subject and Computational Model

2.1. Medical Compound Plunger Pump

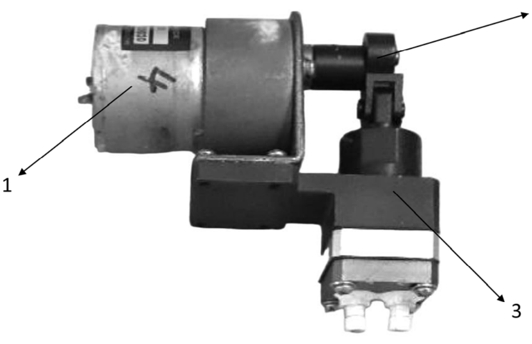

Fig 1. Medical Composite Plunger Pump
1.Power end 2.Transmission end 3.Hydraulic end

Table 1. Main Parameters of the Medical Composite Plunger Pump

Parameter Name	Parameter Value
Pump spindle speed n /(r/min)	100-300
Plunger stroke s /mm	6
Plunger diameter D/mm	16
Crank connecting rod length l/mm	13.8
Crank radius r/mm	3

research subject is a novel reciprocating composite plunger pump, which comprises three main parts: the power end, transmission end, and hydraulic end. The structural diagram of the composite plunger pump is illustrated in Figure 1, while its primary parameters are detailed in Table 1. The output flow rate of the plunger pump is governed by the volumetric variation within the valve chamber of the hydraulic end, which, in turn, is contingent upon the plunger stroke and the frequency of its reciprocating motion. Consequently, precise adjustment of the outlet flow rate of the composite plunger pump can be achieved either by modifying the eccentricity of the crank-connecting rod mechanism to alter the plunger stroke or by regulating the motor's output voltage to adjust the rotational speed and, consequently, the reciprocating frequency of the plunger.

Figure 2 depicts the structural schematic of the medical composite plunger pump. During the startup phase of the pump, when the plunger moves leftward, the pressure within the pump chamber decreases. The valve of the inlet check valve opens due to the pressure difference; driven by the pressure differential, the depressurized air medium inside the chamber draws ambient air into the pump chamber through the inlet check valve.

Conversely, when the plunger moves rightward, the pressure within the pump chamber increases. The valve of the outlet check valve opens in response to the pressure difference, and the pressurized air medium is discharged from the pump chamber through the outlet check valve. Thus, the composite plunger pump completes one operating cycle.

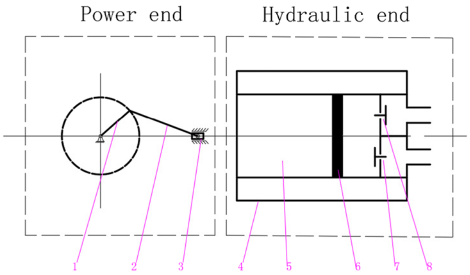

Fig 2. Schematic diagram of the medical composite plunger pump 1.Crank 2.Connecting rod 3.Connecting rod-piston rod connector (bearings, spacers, etc.) 4.Cylinder barrel 5.Pump chamber 6.Plunger 7.Inlet check valve 8.Outlet check valve

2.2. Operating Principle of the Medical Composite Plunger Pump

The micro motor is activated upon energization. Power is transmitted to the connecting rod through the connection between the motor rotor output shaft and the eccentric shaft, thereby driving the plunger to reciprocate within the pump chamber.When the plunger movement causes the pump chamber volume to expand, the air medium under normal temperature and pressure inside the chamber expands, and the internal pressure gradually decreases. The inlet check valve opens under the pressure difference, allowing ambient air to enter the chamber through the inlet check valve.Conversely, when the plunger movement reduces the pump chamber volume, the internal medium is compressed, and the pressure inside the pump chamber gradually rises. The outlet check valve opens under the pressure difference, and the pressurized air medium in the chamber is discharged out of the pump through the outlet valve. After one full revolution of the motor rotor output shaft, the plunger pump is ready for the next working cycle.

2.3. Governing Equations and Turbulence Equations

Atmospheric air at normal temperature and pressure is used as the fluid medium to investigate the flow field inside the medical composite plunger pump. The standard k-ω turbulence model is adopted to conduct unsteady numerical simulations of the internal flow in the pump and valve. The flow in the composite plunger pump satisfies the continuity equation and momentum equation, while the effect of the energy equation is neglected.

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho u_x}{\partial x}+\frac{\partial \rho u_y}{\partial y}+\frac{\partial \rho u_z}{\partial z}=0 \qquad (1)$$

$$\frac{\partial(\rho\vec{u})}{\partial t}+\vec{u}\nabla(\rho\vec{u})=\rho\vec{F}-\nabla P+\mu\nabla^2\vec{u} \qquad (2)$$

In which,ρ is the density of the medium in kg/m³, t denotes time, and ∇ represents the gradient operator.

In this paper, the standard k-ω turbulence model is employed to conduct a transient numerical analysis of the internal flow in the composite plunger pump during the reciprocating motion of the plunger.The k-ε two-equation turbulence model with standard wall functions is selected, which is suitable for flow field simulations of rotating machinery and reciprocating fluid equipment.

Transport equation of turbulent kinetic energy k:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon+Y_M \qquad (3)$$

Transport equation of turbulent dissipation rate ε:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\varepsilon\rho u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad (4)$$

where$u_i$denotes the velocity component in the i-direction with i=1,2,3 ; μ is the kinematic viscosity of the medium; G_k and G_b represent the turbulent kinetic energy produced by the mean velocity gradient and buoyancy, respectively; Y_M stands for the contribution of turbulent fluctuation diffusion induced by compressible flow to the total dissipation rate; σ_k and σ_ε are the turbulent Prandtl numbers corresponding to k and ε, respectively; and,$C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ are empirical constants.

2.4. Dynamic Mesh Scheme

The plunger of the medical composite plunger pump performs periodic reciprocating motion. Since the fluid boundary involves complex moving boundaries, dynamic mesh technology is adopted to simulate the internal flow in the pump. Given that there exists a clearance between the valve seat and the valve core in the initial mesh, with the clearance h between the valve and the valve seat being 0.3 mm, a narrow gap is set between the valve seat and the valve plate to ensure that at least one layer of mesh exists on both the upper and lower sides of the valve plate, thus facilitating the convergence of the numerical calculation.

2.4.1. Dynamic Mesh Update Method

As shown in Fig. 3, the plunger motion zone at the right end of the medical composite plunger pump is a standard cylinder, which is meshed using structured grids. The boundary variation of the fluid domain corresponds to the displacement of the plunger base, and the layering method is adopted for dynamic mesh generation of the plunger zone.As for the inlet and outlet check valve regions, the valve plates move dynamically with the periodic variation of flow rate, resulting in complex geometry; thus, tetrahedral meshes are employed in these regions.The mesh in the inlet and outlet check valve zones is dynamically updated using the spring smoothing method and local mesh remeshing method.[4]

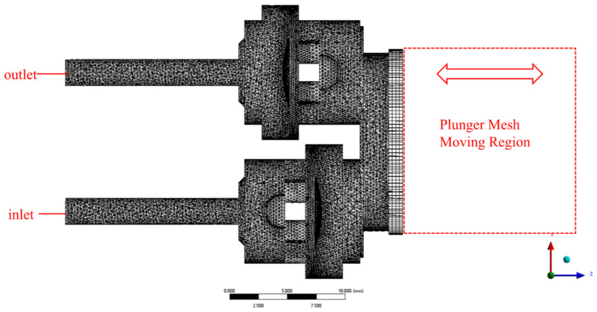

Fig 3. Mesh on the central plane of the computational domain

3. Dynamic Coupling Characteristic Analysis

As shown in Fig. 2, during the start-up phase of the medical composite plunger pump, the rotation of the crank transmits power to the plunger, causing it to move rightward. Driven by the pressure difference, the external fluid medium induces deformation of the inlet check valve, allowing the fluid medium to enter the pump chamber. Meanwhile, the deformation of the check valve alters the fluid flow domain. This phase constitutes a suction process. Similarly, the leftward movement of the plunger corresponds to a fluid-structure interaction process, which requires the solution of the motion equations.

3.1. Plunger Motion Equation

$$v_{piston}=r\omega\left(\sin\phi+\frac{\sin 2\phi}{2\sqrt{1-\frac{r^2}{l^2}\sin^2\phi}}\frac{r}{l}\right) \tag{5}$$

Where r is the crank radius in m;ω is the crank rotational speed in rad/s;ϕ is the crank angle (0°–180°); and l is the connecting rod length in m.

3.2. Fluid-Structure Interaction Interface Condition Equations and Valve Clearance Model

At the fluid-structure interaction interface, the fluid and solid domains must satisfy the displacement continuity condition and force equilibrium condition, with the governing equations given as follows:

$$\begin{cases} d_f=d_s \\ n\tau_f=n\tau_s \end{cases} \tag{6}$$

Where d_f and d_s denote the displacements of the fluid and solid, respectively. Since the fluid has no fixed shape, the displacement at the fluid interface follows that of the solid boundary. τ_f and τ_s represent the stresses at the fluid-solid coupling interface for the fluid and solid, respectively. [5]

The check valve plate moves under the combined action of the fluid pressure difference, the elastic force of the material itself, and the fluid resistance. The differential equation of motion for the valve is established as follows:

$$m\frac{d^2h}{dt^2}=(P_s-P)f_s-k_s(h+h_0)-C\frac{dh}{dt} \tag{7}$$

where: m is the mass of the valve core; P_s is the liquid pressure at the inlet and outlet of the valve chamber; P is the fluid pressure inside the valve chamber; f_s is the force area of the valve core; k_s is the spring stiffness; h is the valve lift;$h_0$is the pre-compression of the valve plate assembly; and C is the overall drag coefficient.

3.3. Flow Field Characteristics Analysis

To investigate the time-dependent flow field distribution characteristics of the medical composite plunger pump, numerical simulations were carried out to obtain the velocity and pressure distributions in the fluid domain within one working cycle. The results reveal the evolution law of the entire fluid domain characteristics during the fluid-structure interaction between the flow field and the check valve core.

3.3.1. Velocity Distribution of the Flow Field

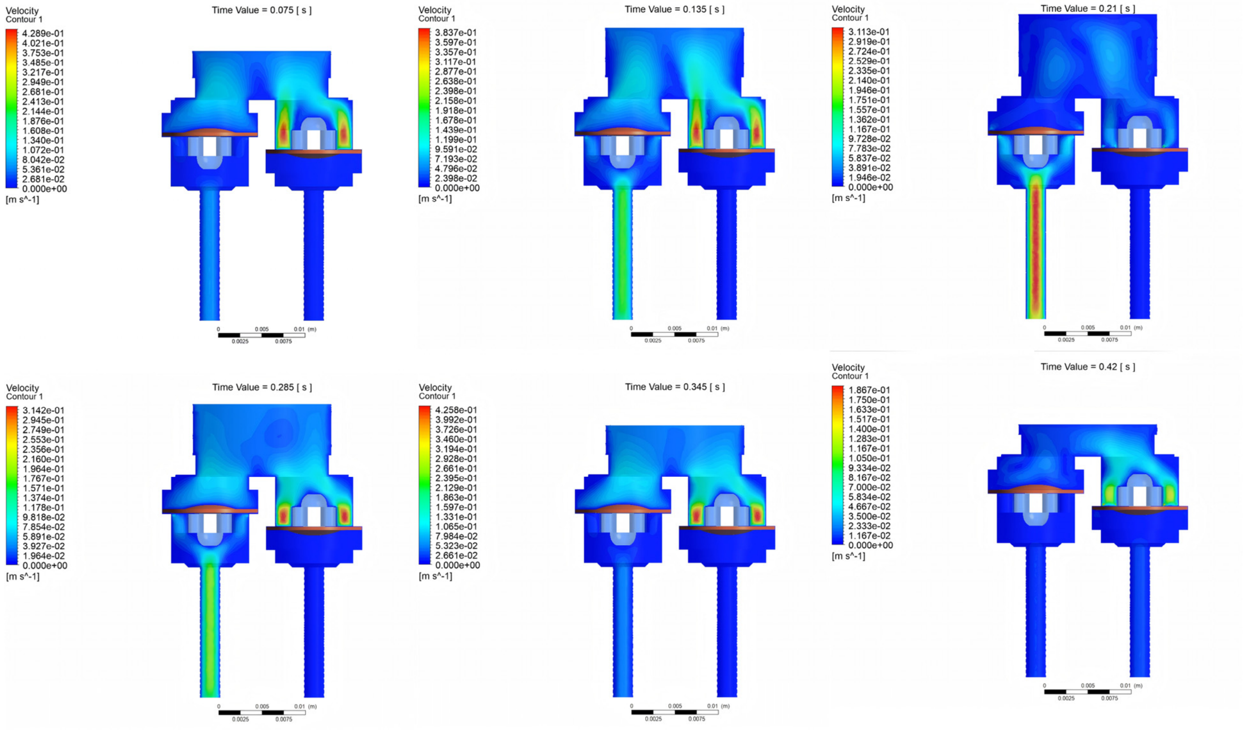

Fig 4. Velocity contour

The flow field model inside the plunger pump is a centrally symmetric structure. Taking the central symmetric plane as the research surface, numerical simulations were performed on the fluid flow in the plunger pump during one reciprocating cycle. The velocity contours of the flow field at different instants are shown in Fig. 4. By observing the flow velocity distributions in the figures, the following conclusions can be drawn:

It can be seen from Fig. 4 that during the opening period of the check valve, high-speed fluid is mainly distributed in the inlet region of the left valve body, especially in the local areas of the valve clearance on the left side of the valve core and the inlet of the fluid domain, showing high flow velocity characteristics. As time goes on, the area of this region gradually decreases in the contour plot.

At 0.21 s in the working cycle, the check valve reaches its maximum lift. The fluid velocity at the inlet of the valve clearance flow field reaches the maximum value of 0.31 m/s, and the fluid flow begins to stabilize. The velocity field inside the pump chamber is relatively uniform, while the velocity distribution at the valve clearance is non-uniform, with the maximum velocity appearing at the opening of the valve plate.

3.3.2. Pressure Distribution of the Flow Field

Figure 5 shows the contour plots of the flow field pressure distribution. It can be seen from Fig. 5 that during the opening period of the inlet check valve, a low-pressure zone is formed in the fluid domain on the pump chamber side of the valve body, and a wide-range pressure gradient variation appears in the fluid region around the passage inlet of the valve clearance.

During the period of 0.135–0.21 s, when the outlet check valve is closing, the minimum pressure in the overall fluid domain inside the pump chamber reaches -35 kPa, and the pressure gradient phenomenon in the inlet region of the valve clearance gradually disappears.

During the opening period of the outlet check valve, a high-pressure zone is formed in the fluid domain on the valve chamber side of the valve body, while the fluid pressure in the valve chamber side of the inlet check valve and the plunger moving surface region remains relatively stable.

During the period of 0.345–0.42 s, when the inlet check valve is closing, the maximum pressure in the overall fluid domain inside the pump chamber reaches 45 kPa, and the pressure gradient phenomenon in the outlet region of the valve clearance gradually disappears.

Within one opening-closing cycle of the two check valves, the pressure distribution and peak value in the flow domain of the inlet check valve generally show a trend of decreasing first and then increasing, while those in the flow domain of the outlet check valve generally show a trend of increasing first and then decreasing. The fluid in the valve clearance flow field exhibits a large-amplitude pressure gradient variation.

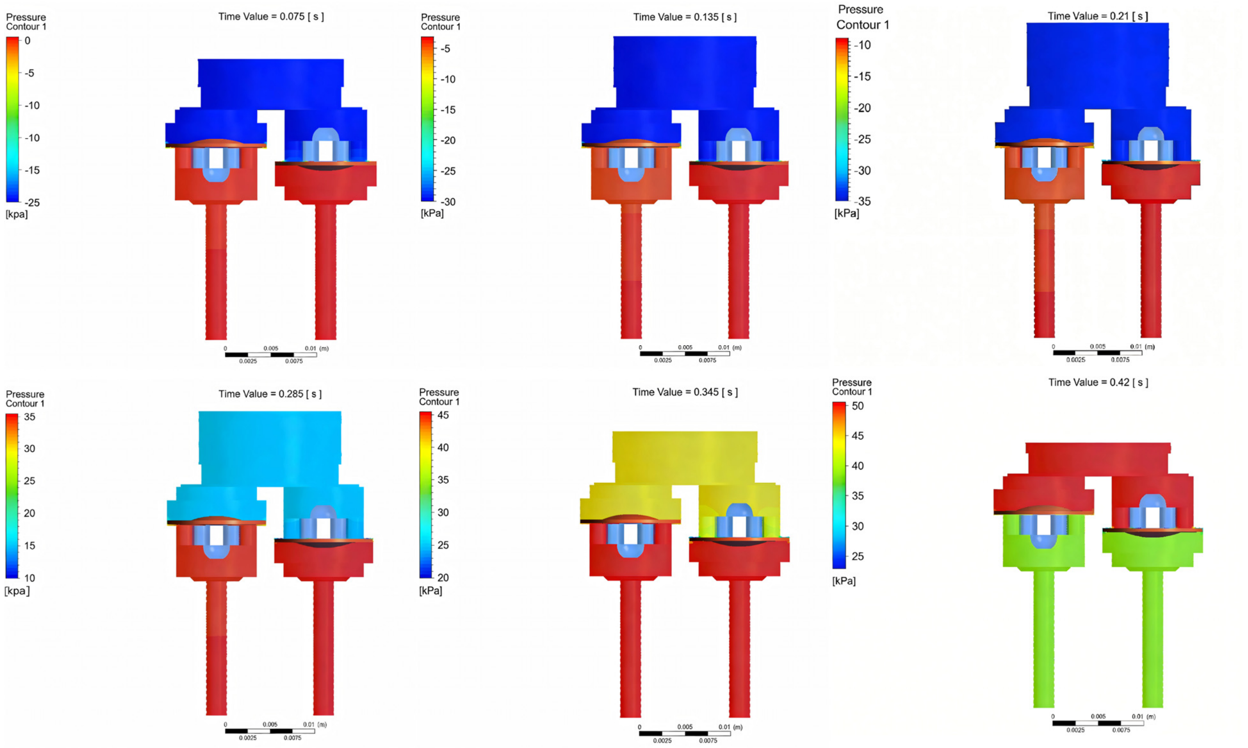

Fig 5. Pressure contour

4. Conclusion

In this paper, the flow rate characteristics of a medical composite plunger pump are investigated using a two-way fluid-structure interaction (FSI) method. A fluid-structure interaction model of the plunger pump is established by employing the dynamic mesh technique in computational fluid dynamics (CFD). The outlet flow rate curve of the medical composite plunger pump under actual working conditions is obtained, and the mechanism of flow rate fluctuation is explained. The reliability of the numerical model is verified by comparison with experimental data. The influences of different model parameters on flow rate characteristics are also analyzed. Based on the above research, the main conclusions are drawn as follows:

(1) The flow rate characteristics of the medical composite plunger pump are analyzed based on numerical simulation results. The results show that the outlet flow rate curve of the plunger pump exhibits a periodic sinusoidal variation.

(2) The pressure loss at the valve port is significantly higher

than that in other regions of the flow field. To reduce pressure loss and improve the efficiency of the medical composite plunger pump, the geometry of the valve port should be optimized.

(3) During one working cycle of the plunger pump, a large pressure difference exists on both sides of the check valve core. In the suction stroke, a low-pressure zone forms at the inlet side of the inlet check valve, with a pressure peak of approximately -39 kPa. In the discharge stroke, a high-pressure zone forms at the outlet side of the outlet check valve, with a pressure peak up to 44 kPa. The fluid in the valve clearance flow field undergoes a substantial pressure gradient variation within one cycle.

(4) At the beginning of the suction and discharge processes of the medical composite plunger pump, delayed closing occurs in the inlet and outlet check valves respectively, accompanied by backflow through the check valves. How to optimize the structure to mitigate this effect will be the key focus of future research.

References

[1] J. Dong, Y. S. Liu, H. Wang, L. J. Wei, and D. F. Wu, "Simulation of Unsteady Flow Characteristics of the Reciprocating Pump Check Valve for High Pressure and High Flow Water Medium," Journal of Fluids Engineering, vol. 144, no. 3, 031206, 2022.

[2] G. R. Wang, L. Zhong, X. He, Z. Q. Lei, G. Hu, R. Li, and Y. H. Wang, "Dynamic Behavior of Reciprocating Plunger Pump Discharge Valve Based on Fluid Structure Interaction and Experimental Analysis," PLOS ONE, vol. 10, no. 10, e0140396, 2015.

[3] J. M. Bergada, S. Kumar, D. L. Davies, and J. Watton, "A complete analysis of axial piston pump leakage and output flow ripples," Applied Mathematical Modelling, vol. 36, no. 4, pp. 1731-1751, 2012.

[4] X. Z. Mi, Q. W. Li, F. Wang, and X. J. Zhao, "Flow field simulation of reciprocating compressor based on dynamic mesh technology," in 2017 IEEE 21st International Conference on Computer Supported Cooperative Work in Design (CSCWD), 2017, pp. 567-572.

[5] Z. J. Gu, C. Q. Bai, and H. Y. Zhang, "Experimental and Numerical Research on Fluid Dynamic Interaction Effects of Reciprocating Pump-Pipeline System," Journal of Pressure Vessel Technology, vol. 145, no. 3, 031401, 2023.